



Development of a Combustor to burn raw producer gas

Srinivasan Dattarajan^{*}, Ramsatish Kaluri, G. Sridhar

Siemens Technology and Services Pvt. Ltd., Research and Technology Center India, 84 Keonics Electronics City, Hosur Road, Bangalore 560100

ARTICLE INFO

Article history:

Received 23 August 2013

Received in revised form 15 April 2014

Accepted 17 April 2014

Available online 10 May 2014

Keywords:

Stirling engine

Producer gas

Combustion

Particulate matter

Tar

ABSTRACT

The development of a combustor, designed to burn raw producer gas that is not subjected to conventional gas cleaning processes, is presented. The combustor had tangential fuel and air inlets that caused a swirling flow inside it in order to facilitate fuel-air mixing and flame stabilization. The mixing and combustion performances of the combustor are characterized herein experimentally and via numerical simulations. Both measurements and simulations indicated good fuel-air mixing, the presence of a stable flame over a range of operating conditions and complete combustion of fuel. Measurement of particulate matter concentration at the combustor exit indicated that the combustor conforms to Indian regulatory norms.

© 2014 Elsevier B.V. All rights reserved.

1. Introduction

Clean energy technologies have been gaining prominence the world over in order to combat Green House Gas (GHG) emissions. Among them, advanced technologies such as gasification are being pursued for fuels such as coal and biomass. Biomass gasification is one such technology that makes economic sense in addition to curbing GHG emissions, especially for emerging economies such as India, China and Brazil. These technologies are suitable for both power generation and heating applications wherein fossil fuels can be easily substituted for.

Biomass gasification essentially comprises the conversion of biomass such as wood, coconut shells etc. into producer gas in an air-starved environment. This gas can in turn be used either for heating applications or be burnt in a gas engine-generator in which electrical power is generated. Producer gas emanates from a biomass gasifier at a temperature of about 500–600 °C, and contains a significant amount of particulate matter (carbon and inorganic particles) and tar (composed of higher hydrocarbons). The typical composition of producer gas from one such reactor design, called an open-top downdraft gasifier and using coconut shells as feedstock, is shown in Table 1. The concentration of particulate matter in the producer gas generated by this gasifier was about 700 mg/Nm³ and that of tar, less than 200 mg/Nm³ [1]. However, this gas cannot be used directly in prime movers such as diesel or gas engines, but requires to be put through an elaborate cooling and cleaning procedure [2,3], which in turn requires several hardware components such as gas cooler, scrubbers, chiller, fabric filter, water treatment plant etc. [1,4]. The steps involved in gas cleaning of a typical downdraft gasifier system are shown schematically in Fig. 1.

The extensive gas cooling and cleaning system requires a large amount of captive power, which reduces the net overall efficiency of the plant. Therefore, from the point of view of reducing captive power consumption, equipment footprint and capital cost, it is desirable to minimize or eliminate the steps involved in gas cleaning/cooling. The approach of directly burning producer gas laden with tar and particulate matter, and using it in conjunction with a Stirling engine, has been developed here. This is expected to improve the overall efficiency.

The Stirling Engine, unlike a gas engine, is an external combustion engine, and can be used to convert heat to mechanical work or electrical power. Being an external combustion engine, it is immaterial how the heat provided to it is produced; this makes it versatile in that it can accept heat from a variety of sources such as solar radiation, combustion of fossil fuels or biomass, and even waste heat from industrial processes. Another advantage of the Stirling engine is that its theoretical efficiency equals the maximum permitted thermodynamically.

The experimental test facility of the present work had a 38 kW Stirling engine installed. This engine had its own burner designed to operate on clean gaseous fuels such as Liquefied Petroleum Gas (Propane – 40%, Butane – 60%). However, this burner was not suitable for burning unclean producer gas as its passages, which were too narrow, would get clogged. This is true of other commercially available gas burners as well. Therefore, there was a need for a burner that could burn producer gas laden with tar and particulate matter. The hot products of combustion of such a burner could be used for electricity generation in conjunction with a Stirling engine; they could also be used for heating and cooling applications. Space heating, steam production and operation of furnaces in agro-based industries are examples of potential applications.

The motivation for undertaking the present development work is the non-availability of burners/combustors commercially for raw producer gas. To the best of the knowledge of the authors, there has been

^{*} Corresponding author. Tel.: +91 80 3313 3634; fax: +91 80 3313 4503.
E-mail address: srinivasan.dattarajan@siemens.com (S. Dattarajan).

Table 1
Composition of raw producer gas.

Major constituent	% by volume
H ₂	15–16
CO	20–21
CH ₄	1.5–1.8
CO ₂	10–12
H ₂ O	3.5–4
N ₂	Balance

no systematic work carried out or reported using raw producer gas as fuel. The development work reported herein aims to address this gap. Design, prototype building and testing with raw producer gas have been reported here for the first time. Critical aspects of combustor performance such as sustenance of a flame without blow-off over a range of operating conditions and compliance with emission norms have been addressed.

2. Approach

The development of a well-instrumented combustor test rig for raw producer gas was quite a challenge. Using existing information in literature, a cyclone combustor was designed. The swirl number [5] targeted was about 4 over the operating range of the combustor. The fact that the paths along which fuel and air entered were off-axis with respect to the combustor yielded the following advantages:

1. Both fuel and air were given a swirling motion in the presence of which they underwent rapid turbulent mixing with each other.
2. This enabled one to operate the combustor at near-stoichiometric conditions, i.e. without introducing a significant amount of excess air.
3. In heating applications, when the combustor is operated under fuel-lean conditions to ensure complete burning, fouling on the walls is accelerated due to the presence of excess air. This problem was avoided due to the effective fuel-air mixing facilitated by swirl.
4. Swirl is known to enhance the flame stability and therefore most suitable for low calorific value gas like the producer gas.
5. Further, swirl enhances the residence time of hot gases in the burner thereby increasing the possibility of complete burning of particulate matter and tar.

A number of configurations were initially identified and Computational Fluid Dynamics (CFD) simulations were carried out for mixing behaviour studies. Among the different configurations, one design was finalized based on criteria such as better mixing, low pressure drop and ease of manufacture. This cyclone combustor and its geometry will be discussed in the next section. Since fuel and air were introduced into the combustor via separate inlets before they burned, combustion was non-premixed. So it was essential for fuel and air to be able to mix effectively and rapidly once introduced into the combustor. The mixing performance of the combustor in the absence of combustion or heat release is studied in this paper with the aid of experimental measurements and numerical simulations, the results of which will be presented in Section 5.

Subsequent to ensuring good mixing behavior, the combustor was commissioned and successfully tested over a range of flow conditions; i.e., raw producer gas from a biomass gasifier was burnt in air inside the cyclone combustor. Measurements of particulate matter concentration indicated conformity with Indian Central Pollution Control Board (CPCB) norms. Numerical simulations to understand the burning characteristics of the combustor were carried out, and their results presented.

3. Experimental setup

Detailed descriptions of the test facility and the cyclone combustor are provided in Subsections 3.1 and 3.2 below. Two broad categories of experiments were performed in the present work – mixing and combustion. The modification done to the test setup in order to perform mixing measurements will be discussed in Subsection 3.4.

The air-fuel ratio λ is defined as the reciprocal of the equivalence ratio [6]. Thus fuel-air mixtures with $\lambda > 1$ are fuel-lean, whereas those with $\lambda < 1$ are fuel-rich. The combustion tests were performed over a λ range of 0.77 to 2.79.

3.1. Test facility

A schematic diagram of the combustor test facility is shown in Fig. 2. Solid biomass such as coconut shells was gasified in the biomass gasifier resulting in the production of producer gas, the particulate matter in which was partially removed in a cyclone separator. The concentrations of particulate matter and tar in the gas at the exit of the cyclone separator were about 400–450 mg/Nm³ [7] and 200–250 mg/Nm³, respectively. The producer gas exiting the cyclone separator could be directed into either of two flow paths. Using the first flow path, the producer gas could be introduced into the cyclone combustor via its fuel inlet. The second flow path led to a gas cleaning system and then either a gas engine or flare shown in Fig. 1. Since the cyclone combustor was located at a distance from the gasifier system, the hot producer gas from the exit of cyclone separator was conveyed using a long insulated pipe, resulting in drop in temperature of producer gas at the inlet of the cyclone combustor. The gas flow paths were operated one at a time using pneumatic valves V_1 and V_2 .

The exhaust products of the cyclone combustor passed through a recuperator, a shell and tube heat-exchanger, in which they pre-heated air drawn from the surroundings. This pre-heated air was subsequently directed to the air-inlet of the cyclone combustor. The flowrate of air was regulated using an A/F (air-fuel ratio) controller, which for a given flowrate of gas into the fuel inlet, regulated the air flowrate until a desired fuel-air flowrate ratio was achieved. A blower at the downstream end of the test facility created the requisite suction to draw gases into the cyclone combustor, and product gases out of it. The cyclone combustor is described in the next subsection.

In order to reduce the temperature of the flue gases before they reached the recuperator, they were diluted with ambient air. The temperature limits on the recuperator inlet and the blower inlet arising from material constraints necessitated the use of dilution air under certain operating conditions.

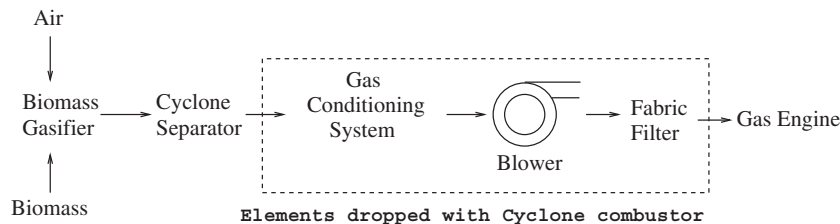


Fig. 1. Conventional approach to extraction of electrical power from biomass at sub-megawatt power level.

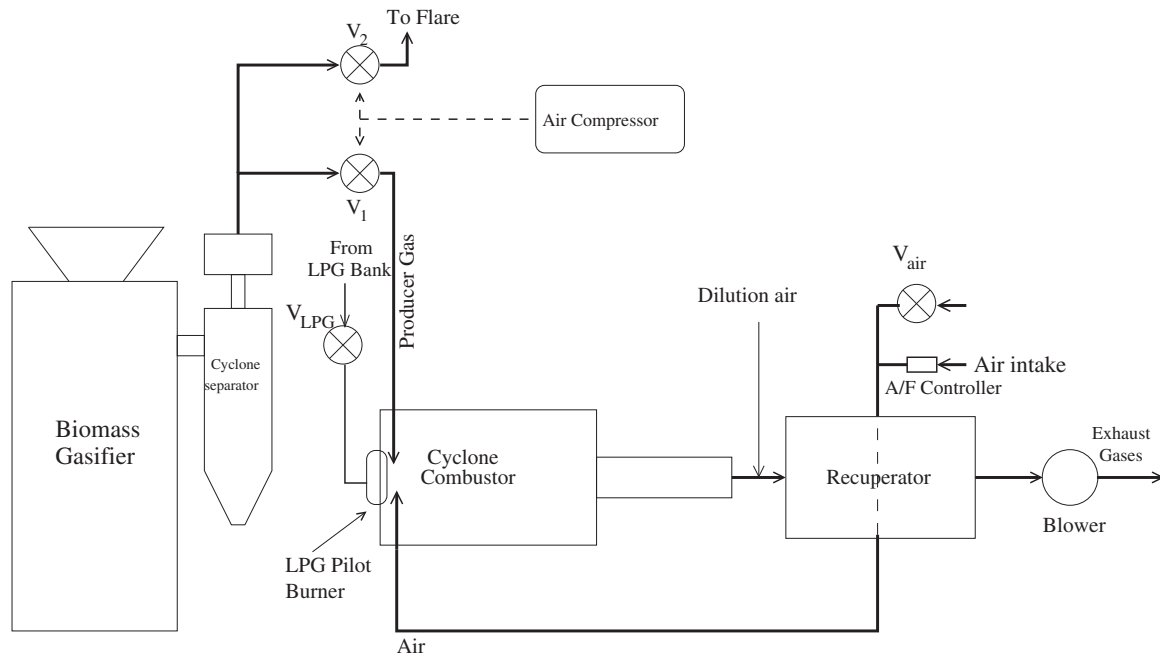


Fig. 2. Schematic diagram of the combustor test facility.

The entire hot gas path, starting from the exit of the biomass gasifier, up to the inlet to the blower, was fabricated using Stainless steel (SS304) and further insulated with multiple layers of alumino-silicate blanket, which has a thermal conductivity of 0.11 W/m-K.

3.2. Cyclone combustor

The cyclone combustor was the centerpiece of the combustor test facility. It was designed to directly burn fuel laden with particulate-matter and tar, without the need for elaborate gas cleaning apparatus. The combustor, shown schematically in Fig. 3, was fabricated using Stainless Steel (SS310), which allowed a maximum operating temperature of about 1300 °C. Major dimensions of the combustor are shown in Fig. 3. The fuel and air inlet dimensions were 100 mm × 60 mm.

The fuel and air inlets did not have any narrow passages that could result in blockage due to adherence of particulate matter or tar. The inlets were placed such that both fuel and air entered the combustor tangentially, thereby creating a swirling flow within. The swirl is what lent the cyclone combustor its name. The major dimensions of the combustor such as diameter and length were arrived at from separate non-reacting flow CFD studies, which are not described here. As mentioned earlier, this effort involved conducting CFD simulations on a number

of configurations for predicting mixing behavior. Among the different configurations, the design described here was finalized based on criteria such as better mixing, low pressure drop and ease of manufacturing.

A pilot burner fueled by Liquefied Petroleum Gas (LPG) was installed at the upstream end of the combustor, as shown in Fig. 3. The pilot flame was incorporated into the design for two purposes: (a) to facilitate ignition of producer gas (for a maximum duration of 10–15 minutes) as it was introduced at the time of start-up of the combustor (b) to act as flame stabilizer in the event of flame blow-off under fuel-lean conditions. However, in the experiments performed, there never was a situation wherein flame blow-off was seen even while operating the combustor under fuel-lean conditions with λ as high as 2.79.

3.3. Instrumentation and data acquisition

In the mixing tests, the following instrumentation was in place. Thermocouples of R-type supplied by Tempsens Instruments were used to measure gas or flame temperatures at various locations within the combustor, as shown in Fig. 4. Specifically, three thermocouples were placed along the combustor centreline at various axial locations, two at a radial distance of 2/3 combustor radius from the combustor wall, and three at a radial distance of 1/3 combustor radius from the

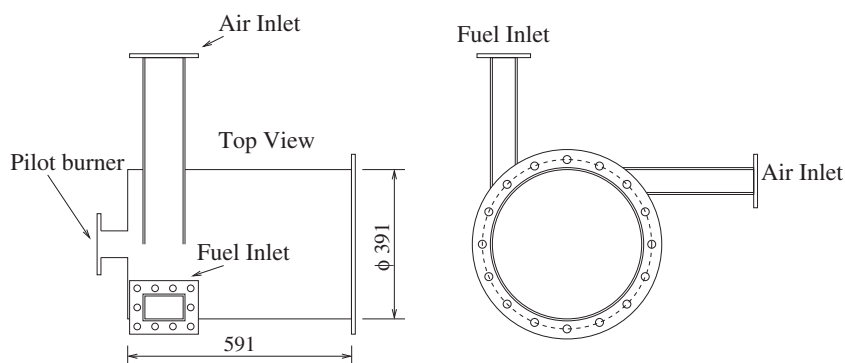


Fig. 3. Schematic diagram of the cyclone combustor. Dimensions shown are in mm.

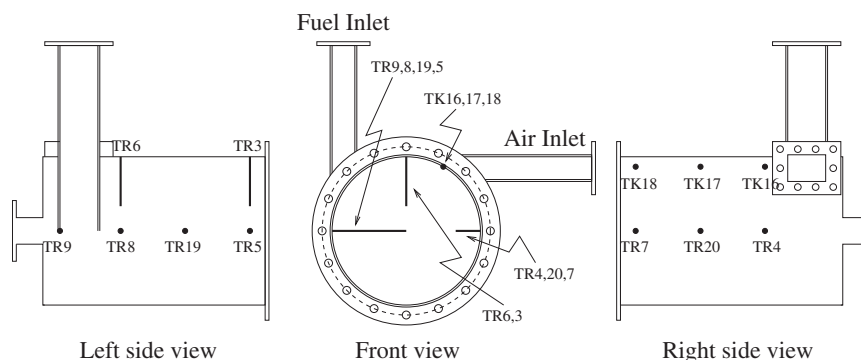


Fig. 4. Locations of thermocouples within the combustor. Thermocouples TR3 and TR6 are not shown in the right side view for clarity.

wall. Thermocouples of K-type were used to measure surface temperatures on the combustor wall, and gas temperatures at the inlets to the combustor. Thermocouples installed outside the combustor are not shown in Fig. 4. Similarly, K-type thermocouples were used to measure the temperature drop across the recuperator and exit of the blower of the test rig shown in Fig. 2.

Pressures were measured at various locations within the test setup using piezoelectric pressure transducers manufactured by IRA Pvt. Ltd., in order to quantify system pressure drop and to aid the measurement of flowrates. Flowrates of producer gas and air entering the combustor, and that of dilution air, were measured using calibrated orifice plate meters placed in the respective flow paths. The system pressure drop, i.e. the drop in pressure across the exit of the biomass gasifier and the recuperator exit, was an indicator of the amount of captive power needed to be supplied to the blower.

To assess the completeness of combustion, concentration of oxygen gas was measured at the combustor exit using a gas analyzer made by Kane equipment. In addition to the above, concentrations of gaseous pollutants, viz. NO_x and CO, were also measured and periodically recorded by the same gas analyzer. Since one of the key features of the combustor developed here was to directly burn tar and particulate matter present in producer gas, it was essential to quantify the concentration of particulate matter at the exit of the combustor. This was done using a United States Environmental Protection Agency (US-EPA)

Method-5 [8] iso-kinetic particulate measurement system supplied by Apex Instruments.

The operating sequence of the experiments was coordinated by a Programmable Logic Controller (PLC) made by Siemens, which had several safety features built in. The PLC controlled the sequencing of opening/closure of pneumatic valves V_1 and V_2 , start-up of the pilot LPG burner and switching on the suction blower. Potentially unsafe events such as extinction of the flame, excessive delay in ignition, pressure build-up etc. were programmed in the PLC to trigger alarms and initiate system shutdown. Time-resolved measurements of temperature and pressure were continually captured and recorded by a data acquisition system supplied by National Instruments.

3.4. Modification to the test setup

The most obvious way to characterize fuel-air mixing inside the combustor would have been to introduce producer gas and air into the combustor without igniting or burning them, and to measure the concentration of a representative species such as CO or O_2 at various locations within the combustor. It would be unsafe to create an explosive mixture of fuel and air, without burning it, in the combustor and the piping leading to the exhaust. Hence the test setup was modified as follows. The modification is shown schematically in Fig. 5.

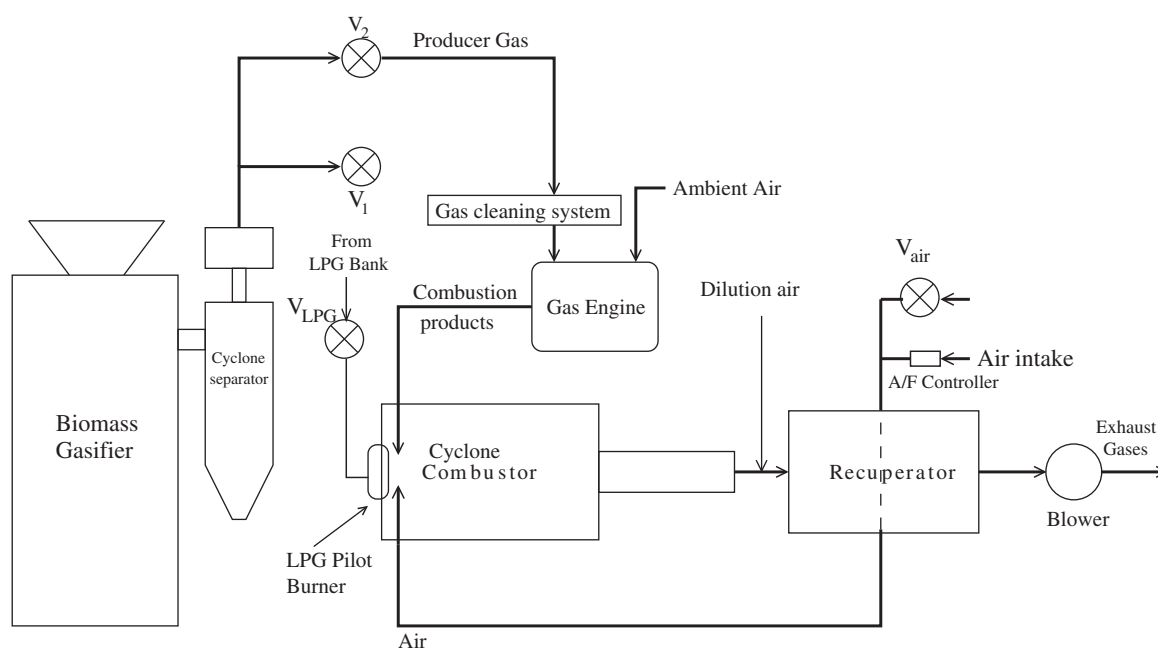


Fig. 5. Modifications to the test facility in order to characterize mixing.

The producer gas output by the biomass gasifier was directed to the second flow path alluded to in Section 3.1; i.e. passed through a gas cleaning system, wherein any particulate matter and tar present were removed. Subsequently, the clean producer gas was burnt with air in a gas engine. The hot products of combustion from the gas engine were directed into the fuel inlet of the cyclone combustor. It was assumed that the mixing behavior of the combustion products replicated that of producer gas. Ambient air was pre-heated in a manner described in Section 3.1 before being introduced into the combustor's air inlet. The evolution of temperature within the combustor in such a test was taken to be a representative of the fuel-air mixing characteristics within the combustor. The mixing results reported in Section 5.1 pertain to the operation of the test facility with this modification in effect.

3.5. Combustion tests

The combustor was started by switching on the LPG pilot flame for about 3–5 minutes for initial heating. Later, the producer gas supply was commenced and the LPG flame was continued for another 10–12 minutes. After the pilot burner was switched off, the flame of producer gas in air was seen to be stable. The typical duration of a single experiment was about 2 hours, which was necessitated by the high thermal inertia of the test setup.

3.6. Test matrix

3.6.1. Mixing measurements

Three runs were performed to characterize mixing within the cyclone combustor. Table 2 shows the combustor inlet conditions for each run. The subscripts hot and cold pertain to the fuel and air inlets, respectively, whereas the symbols $\dot{m}$ and T denote mass flow rate and temperature, respectively. Note that even though ambient air was pre-heated before being directed into the combustor's air inlet, it is referred to as the 'cold' stream in Table 2, meaning simply that it was cold with respect to the hot combustion products entering the fuel inlet.

Each run comprised setting the mass flow rates and temperatures at the fuel and air inlets to one of the sets of values in Table 2. The measurements of flow quantities discussed in Section 3.3 were logged at 1 Hz in the data acquisition system. Each experiment was performed for 10 minutes after all temperatures attained steady-state values. These steady values are reported as measured temperatures in the Results section.

3.6.2. Combustion tests

Two sets of reacting flow tests were conducted. The focus of the first, tabulated in Table 3, was to commission the combustor. The second, tabulated in Table 4, was focused on carrying out the complete set of measurements planned, including those of particulate emissions. The λ values in the tests listed in Tables 3 and 4 ranged from 0.77 to 2.79.

4. Numerical method

The mixing of the hot and cold gas streams, as well as the combustion of producer gas in air, described in the previous section, were modeled using the commercially available CFD modeling software Fluent. The Reynolds Averaged Navier–Stokes (RANS) equations

Table 2
Test conditions.

No.	$\dot{m}_{hot}$ kg/hr	$\dot{m}_{cold}$ kg/hr	T_{hot} °C	T_{cold} °C
1	111	139	292	184
2	106	176	287	178
3	113	203	172	113

Table 3
Test matrix 1 for the combustion experiments.

No.	$\dot{m}_{fuel}$ kg/hr	$\dot{m}_{air}$ kg/hr	T_{fuel} °C	T_{air} °C	λ
1	52.2	75.4	302	304	1.21
2	59.4	92.5	360	539	1.31
3	52	104.73	309	516	1.69
4	57.18	72.34	338	461	1.06

governing the turbulent flow of fluid inside the combustor were solved numerically on a computer.

The modifications made to the test setup to carry out mixing experiments were described in Section 3.4. In the numerical simulations of fuel-air mixing, air at a suitable temperature was introduced into the combustor in lieu of producer gas in order to simplify the computations. It was assumed that the mixing behavior of air replicated that of producer gas. Note that the molecular weights of the combustion products of the gas engine, producer gas and air are respectively, 29.7, 25.2 and 28.8 g/mol.

To facilitate numerical simulations of the flow inside the combustor, its full scale 3-D solid model was created on a computer, which is shown in Fig. 6(a). A few planes along and perpendicular to the combustor axis were created, and is shown in Fig. 6(b). Flow quantities of interest in these planes will be presented in the Results section. The following subsections provide details of the numerical method.

4.1. Computational grid

The continuous space inside the combustor was discretized into finite-sized tetrahedrons in order to enable numerical simulations. These tetrahedral cells are collectively called a mesh. The mesh was generated using the commercially available software ICEM, which placed fine cells near the combustor walls to resolve the boundary layers there, and progressively grew the cells inwards.

Since the cells were not placed at regular spatial intervals, the mesh is said to be unstructured. The cross-sectional view of the mesh in the mid-x plane (see Fig. 6(b)) is shown in Fig. 7. The mesh used herein had 0.7 million tetrahedral cells.

4.2. Turbulence model

The RANS equations, which are obtained by temporally averaging the Navier–Stokes equations governing fluid flow, don't form a complete system of equations. Rather, additional unknowns introduced by the process of time-averaging need to be empirically modeled. For the mixing simulations, the following empirical turbulence models available in Fluent were applied, and predictions from them compared with each other, and with experiment: (i) Realizable $k - \epsilon$ model (ii) $k - \epsilon$ RNG model and (iii) Reynolds Stress Model (RSM). Detailed descriptions of each of these models can be found in standard texts on Turbulence [9].

Table 4
Test matrix 2 for the combustion experiments.

No.	$\dot{m}_{fuel}$ kg/hr	$\dot{m}_{air}$ kg/hr	T_{fuel} °C	T_{air} °C	λ
1	75	136	183	572	1.53
2	72	126	159	486	1.46
3	64	123	166	550	1.62
4	42	66	194	580	1.33
5	43	140	126	491	2.79
6	57	52	155	475	0.77
7	82	102	231	610	1.04

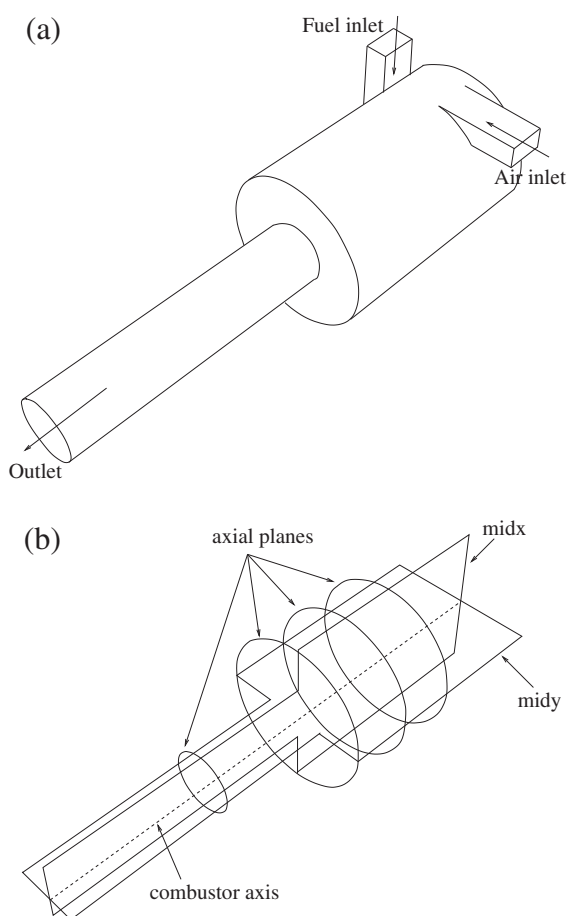


Fig. 6. Computational model of the cyclone combustor. A 3D solid model of the combustor was created to aid numerical simulations. Shown here are the (a) Iso-metric View, and (b) Cut-section planes in various locations, in which various flow quantities will be presented in the Results section.

As will be discussed in the Results, one of the above three turbulence models was down-selected based on the mixing results, and applied to the combustion simulations.

4.3. Combustion model

The combustion of producer gas in air was modeled within Fluent using the non-premixed combustion model wherein equations for the mixture fraction and its variance were solved. A detailed description of this approach can be found in standard texts on turbulent combustion [10]. Equilibrium chemistry was assumed, meaning that reactions reached equilibrium at time-scales much shorter than the flow time-scale. Equations for mixture fraction and its variance were solved in addition to the Navier Stokes equations.

4.4. Boundary conditions

In the CFD simulations, the turbulent flow within the cyclone combustor was treated as statistically stationary, meaning that bulk flow

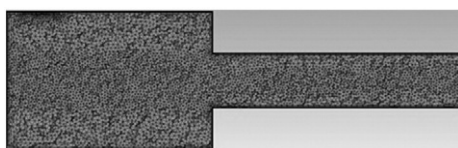


Fig. 7. Cut-section view of the tetrahedral mesh of the cyclone combustor in the mid-x plane.

quantities did not evolve in time. Hence no initial conditions needed to be specified.

4.4.1. Mixing simulations

Boundary conditions were specified within Fluent as follows. The mass-flow rates at the fuel and air inlets were set to fixed values listed in Table 2. The pressure at the exit of the flow domain was set to a constant value of 1 atm on the combustor axis. A radial equilibrium pressure distribution was applied to the rest of the exit plane. The turbulence levels at the combustor inlets were set to 10%, and enhanced wall functions were used to model wall turbulence.

In the initial simulations aimed at studying the effect of turbulence model on mixing, the combustor walls were treated as adiabatic. In all subsequent simulations, Nusselt number correlations [11–13] were used to obtain wall heat transfer coefficients. These coefficients were used in conjunction with the thermal conductivity of the insulation layer discussed in Section 3.1 in order to obtain effective wall heat transfer coefficients.

4.4.2. Combustion simulations

The combustion simulations were conducted on the test matrix in Table 3. Wall heat transfer was modeled as described in Section 4.4.1. In addition to quantities specified therein, temperatures of fuel and air at their respective inlets were specified. The mixture fractions at the fuel and air inlets were set to 1 and 0, respectively.

5. Results

The results pertaining to mixing and combustion studies are presented separately.

5.1. Mixing

5.1.1. Experimental Measurements

The steady values of temperature at the locations discussed in Section 3.3 are presented in Fig. 8 for each of the test conditions listed in Table 2. The direction of bulk flow in that figure is from left to right. Upstream in the combustor, near the fuel and air inlets, a significant variation in temperature is seen with radial distance from the combustor axis. This is because gases entering from the fuel and air inlets were at different temperatures. As the swirling flow evolved along the combustor axis, the temperature difference subsided, and was virtually absent close to the combustor exit.

The nearly uniform temperatures near the combustor exit indicated that the gases entering from the fuel and air inlet underwent effective turbulent mixing for the range of conditions in Table 2. One of the objectives of creating a swirling flow in the combustor was to facilitate turbulent mixing. The data here validate that objective.

5.1.2. Numerical simulations

The numerical simulations provide a far more detailed picture of the flow field than the experimental measurements. As pointed out in Section 4.4, the walls of the combustor were treated as adiabatic initially, wherein all the turbulence models in consideration were applied. Contours of temperature in the mid-x plane from those numerical simulations are shown in Fig. 9.

For each of the test conditions in Table 2, it can be seen that the realizable $k - \epsilon$ and the $k - \epsilon$ RNG models predict near instantaneous mixing of gases entering from the fuel and air inlets. This is evident from the uniform temperature attained throughout the flowfield, except very close to the fuel and air inlets.

The RSM on the other hand, predicted a more gradually evolving temperature field along the axis of the combustor. This is consistent with what one would expect in a swirling flow, and also from the experimental data in Fig. 8. Also, the RSM method is suggested to be a superior method for simulation of turbulent swirling flows in a number of

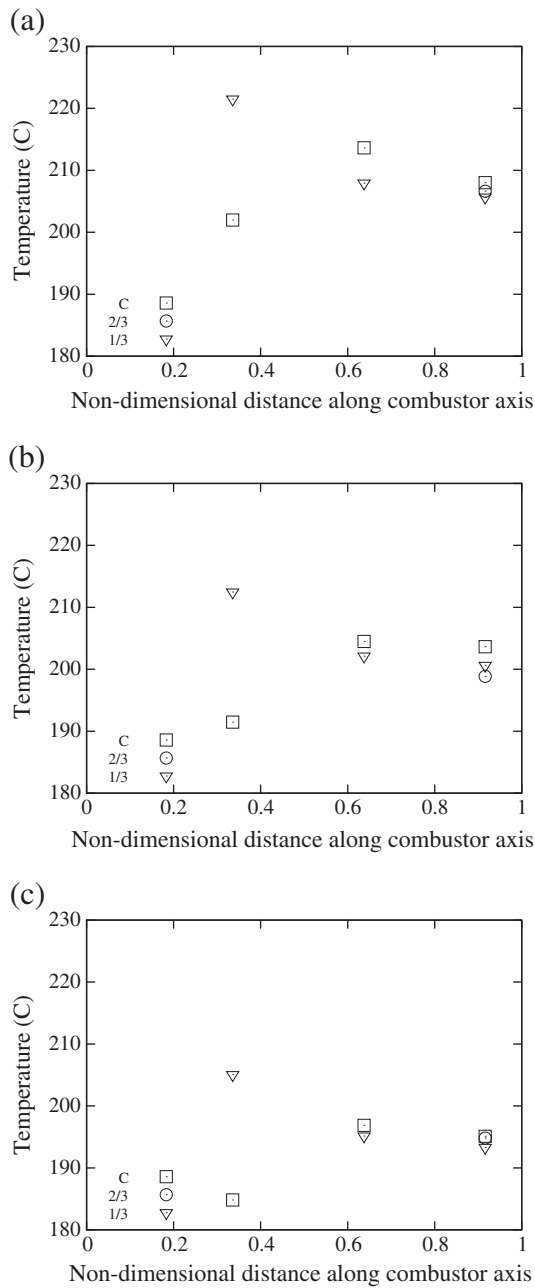


Fig. 8. Measurements of temperature within the cyclone combustor. The labels C, 2/3 and 1/3 in the legends denote radial location of the measurement being at the centerline, 2/3 combustor radius from the wall, and 1/3 radius from the wall, respectively. Subfigures (a), (b) and (c) pertain, respectively, to conditions 1, 2 and 3 in Table 2.

studies [14–17]. For these reasons, the RSM was deemed to be the model more suited to the simulation of the flow within the combustor.

In subsequent simulations, only the RSM was used as the turbulence model. Convective heat transfer at the combustor walls was modeled as described in Section 4.4. Prior to conducting exhaustive simulations, grid independence studies on the computational mesh described in Section 4.1 were performed. Simulations were performed on meshes ranging from 0.4 million to 1.1 million cells in size, and no appreciable difference was seen in a range of flow quantities. A mesh 0.7 million cells in size was deemed adequate for these simulations.

Contour plots of various flow quantities, viz. axial and tangential velocities, static pressure and temperature, from those simulations are shown in Fig. 10 for the first condition in Table 2. The axial velocity contours in Fig. 10(a) show the recirculation region formed because of the

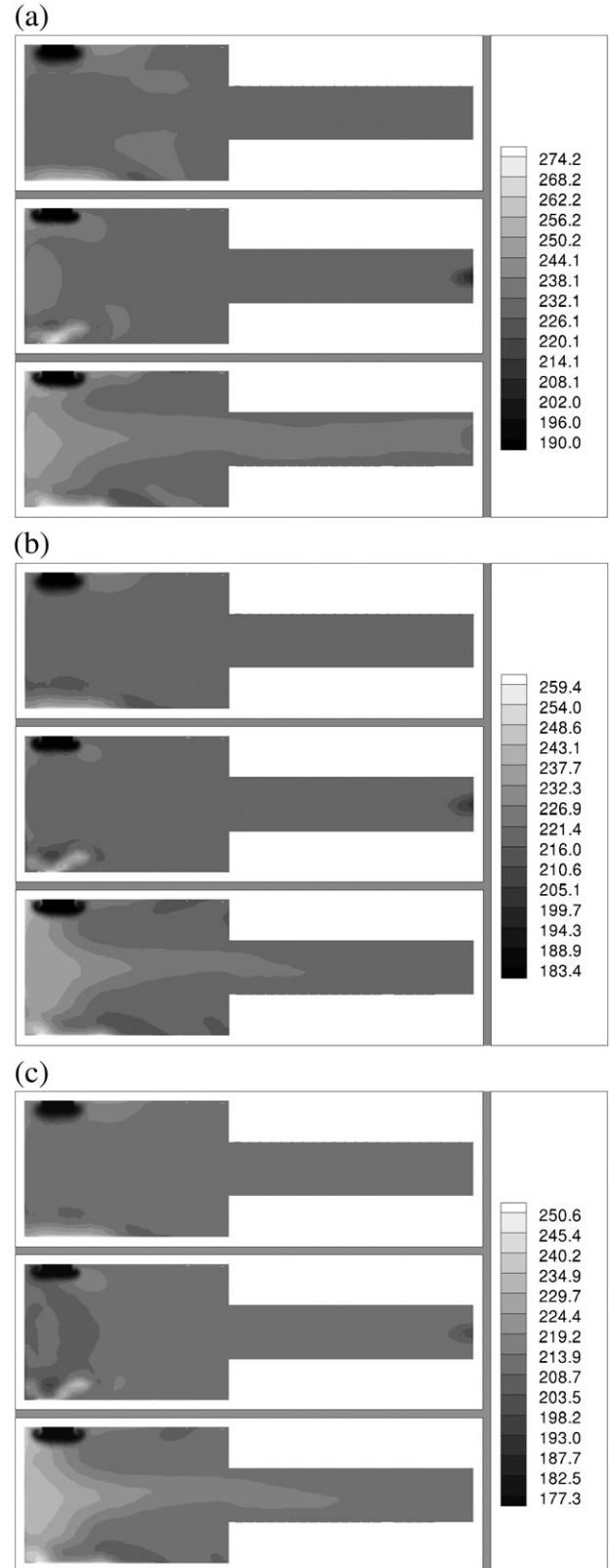


Fig. 9. Contour plots of temperature (°C) in the mid-x plane. Results for test conditions 1, 2 and 3 in Table 2 are shown in (a), (b) and (c), respectively. The three contours in each subfigure are results obtained using the realizable k-ε, k-ε RNG and RSM turbulence models, in that order.

sudden area change at the combustor exit, and the attendant reverse flow. The tangential velocity contours in Fig. 10(b) show the gradual decay of swirl as one progresses along the combustor axis. The radial

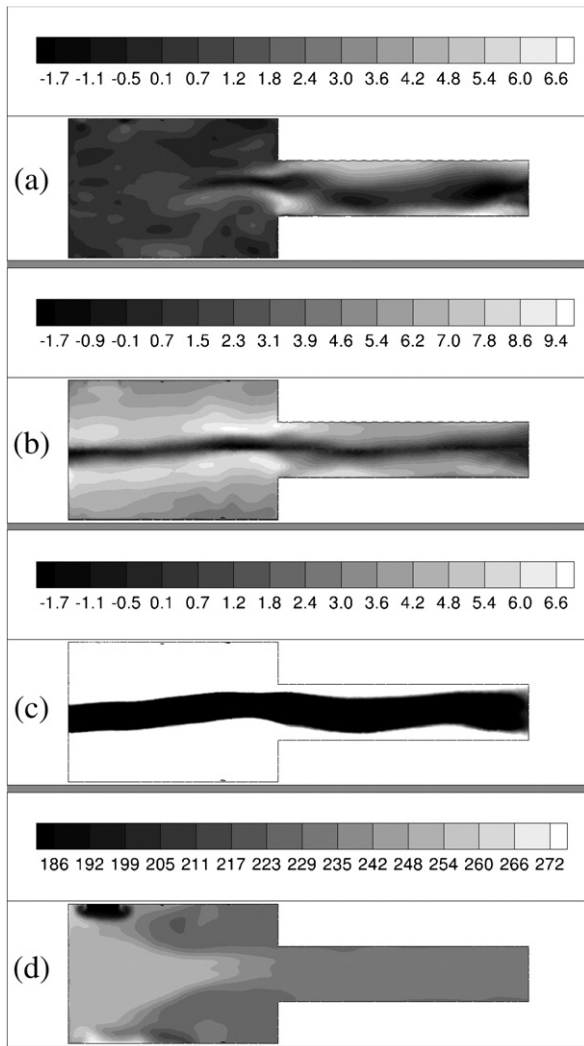


Fig. 10. CFD predictions in the mid-x plane using RSM for test condition 1 in Table 2. Contours of (a) Axial velocity, m/s, (b) Tangential velocity, m/s, (c) Static pressure, Pa, and (d) Static temperature, °C, are shown.

pressure gradient evident from Fig. 10(c) provided the centripetal force required to sustain a swirling flow within the combustor. As in Fig. 9(a), the temperature contour in Fig. 10(d) shows gradual mixing of fuel and air within the combustor.

5.1.3. Comparison of experiments and CFD

In Section 3.4, it was pointed out that in the experiments, the products of combustion of a gas engine were used in place of producer gas, and in the simulations, heated air performed that role. The turbulent mixing characteristics of both gases with air were expected to be similar to that of producer gas, owing to their comparable molecular weight.

In Fig. 11, the temperatures predicted using the RSM turbulence model in CFD are compared to those experimentally measured. The uncertainties of 1% in experimental data approximately equal the size of the symbols used to represent them. Along the centreline of the combustor, the CFD predictions exceed the measured values by over 40 °C far upstream, but this difference subsides as one moves downstream along the combustor. In fact, CFD temperature values at all radial and axial locations exceed the corresponding measurements. Near the exit of the combustor, temperatures at all radial locations approached a common value due to turbulent mixing, though this value is higher for CFD than for experiment.

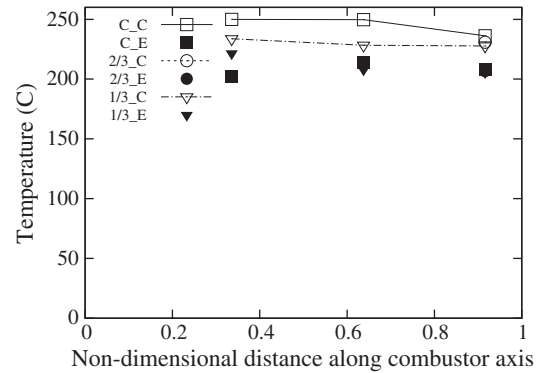


Fig. 11. Comparison of experimentally measured temperatures to those predicted using RSM in CFD for test condition 1 in Table 2. In the legend, C, 2/3 and 1/3 signify, respectively, on the combustor axis, 2/3 combustor radius from the wall and 1/3 combustor radius from the wall. E and C denote experiment and CFD, respectively.

There are two possible reasons for CFD predicting temperatures higher than those measured. The first is the effect of heat capacity. Recall that in the experiments, the products of combustion of a gas engine were introduced into the fuel inlet of the cyclone combustor, whereas in the CFD simulations, air at the same temperature was. The mixture of air and the combustion products of the gas engine, in the proportion in the first condition of Table 2, had a heat capacity of 1085 J/kg-K at 230 °C, the approximate temperature at the combustor exit. At the same temperature, air has a heat capacity of 1038 J/kg-K, which is 4.5% lower. Consequently, the mean temperature at the exit of the cyclone combustor predicted by CFD should be expected to be about 22 °C higher than that measured in the experiments. Fig. 11 suggests that that is indeed the case.

Secondly, the thermocouples were expected to undergo some radiative heat loss due to which the values read off are lower than the actual gas temperatures. This had not been corrected for; however at the temperatures in question, this effect is expected to be small.

5.2. Combustion

5.2.1. Experiments

The cyclone combustor was commissioned and producer gas laden with tar and particulate matter directly burnt with air in it following the steps outlined in 3.5. Fig. 12(a) and (b) show the evolution of temperature near the combustor exit at λ values of 1.04 and 2.79, respectively. These pertain to conditions 7 and 5, respectively, shown in Table 4. The temperature values plotted in Fig. 12 are raw acquired data. Temperature correction for radiation will be discussed in Section 5.2.3, and applied subsequently. The locations of the thermocouples listed in the plot legends in Fig. 12 can be looked up in Fig. 4. The drop in temperature about 1000 seconds into the experiment in these figures resulted from the pilot burner being switched off, and the producer gas flame being allowed to sustain on its own. It can be seen in Fig. 12 that the combustor took a long time to attain steady state. This was indicative of the high thermal inertia of the combustor setup. Moreover, introduction of greater excess air reduced the flame temperature, and thereby the time taken to attain steady state. Hence the temperature profiles in Fig. 12(b) appear flatter than those in Fig. 12(a). The steady temperatures attained at λ values of 1.04 and 2.79 appear to be over 200 °C less than the respective adiabatic flame temperatures of 1866 and 1023 °C. This can be attributed to wall heat loss and radiative heat loss from the thermocouples. The peak measured flame temperature across all experiments was 1378 °C, and was recorded during the experiment at $\lambda = 1.04$.

Overall the combustor operation was stable over the range of λ values tested. There was no event of flame blow-off in the limited set of experiments that were carried out. But due to metallurgical

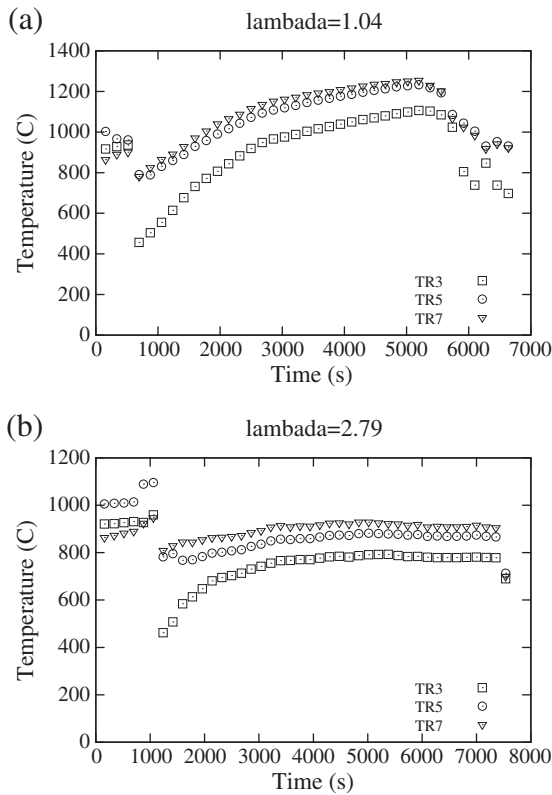


Fig. 12. Time-evolution of temperature at conditions (a) 7 and (b) 5 in Table 4.

constraints, the surface temperature of the combustor was not allowed to exceed 1250 °C. As a consequence, testing of the combustor for longer duration close to stoichiometric conditions was not feasible.

The total pressure drop across the system including the combustor and recuperator was not more than 800 Pa. This result goes to prove that the cyclone combustor incurred low pressure drop and consequently low input power for operations.

Concentrations of particulate matter in the flue gas are shown in Fig. 13. It is clearly seen that increasing λ has the effect of reducing the particulate matter concentration. Although, close to stoichiometric conditions, the hotter flame could burn particulate matter and tar better, the excess air at higher λ had the effect of diluting the flue gases and driving down the particulate matter concentration. It should be noted that the concentrations plotted in Fig. 13 are well below the

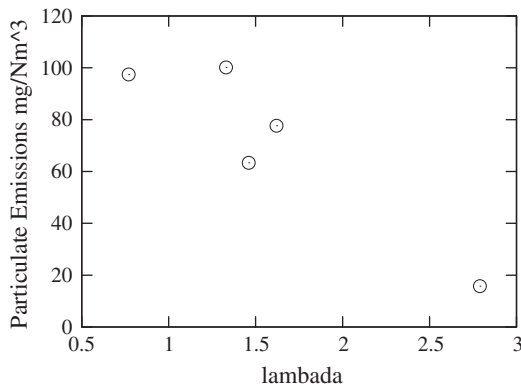


Fig. 13. Particulate matter concentration (mg/Nm³) in the flue gases measured downstream of the combustor.

norms of the Central Pollution Control Board (CPCB) of India [18], which require the particulate concentration in the exhaust of a machine producing up to 210 MW power to be less than 350 mg/Nm³.

Furthermore, the highest measured NO_x across all conditions listed in Tables 3 and 4 was 130 ppm. This is below the CPCB norm [18] of 150 ppm for existing power plants.

5.2.2. Numerical simulations

Contours of temperature and axial-velocity in the mid-x plane from numerical simulations at the inlet conditions in Table 3 are shown in Figs. 14 and 15 respectively. The exit pipe downstream of the combustor was lengthened in these simulations to allow for burning outside the combustor. At $\lambda = 1.06$, the combustion relied heavily on effective fuel-air mixing to complete. Therefore the flame is seen to tongue out of the combustor and continue burning outside in Fig. 14(a). As the proportion of excess air increased in Fig. 14(b), (c) and (d), the flame was progressively drawn into the combustor, and can be seen to be contained within it in Fig. 14(d) at $\lambda = 1.69$.

The fact that the flame was drawn progressively into the combustor and that more complete combustion was attained at the combustor exit with increasing λ might tempt one to operate the combustor at higher λ values. However, there are two disadvantages to that - (a) Owing to the excess oxygen in the flue gases at high λ , and the fact that the flue gases themselves are at a high temperature, the duct work conveying them is susceptible to corrosion, and (b) The excess air that doesn't participate in combustion still has to be pumped, adding to the captive power consumption. Hence it is desirable to operate as close to stoichiometric conditions as possible.

The axial velocity contours in Fig. 15 clearly show back-flow regions ahead of the sudden area change. The higher volumetric flowrates at higher λ owing both to the higher total flowrates (see Table 3) and more complete burning inside the combustor result in higher axial velocities in the exit pipes in Fig. 15(c) and (d).

The exit pipe downstream of the combustor was initially one combustor-length long, e.g. in Fig. 10. For a number of simulations with this geometry, reverse flow was observed near the exit of the flow domain, as in Fig. 10(a). Hence the exit pipe was lengthened to three combustor lengths to allow for better evolution of the flow after it exited the combustor. Nevertheless, in certain cases, e.g. Fig. 15(c) and (d), regions of reverse flow were still seen near the end of the exit pipe. This could be an effect of swirl in the bulk flow in the exit pipe. In order to sustain a swirling flow, it is essential to have a finite radial pressure gradient, resulting in a low pressure region along the combustor axis. Consequently fluid from the exit region could have been drawn towards there, creating pockets of back flow. In other words, the observed back flow need not merely be a numerical artifact resulting from the pressure boundary condition at the exit, but has physical significance as well.

5.2.3. Temperature correction for radiation

The diameter of the thermocouple stem was chosen to be 4.5 mm to balance two needs - to get an accurate temperature reading without high radiative heat loss, and to have a structurally sturdy stem that could withstand the harsh conditions inside the combustor. Owing to the high temperatures prevalent inside the combustor, radiative heat loss was unavoidable. The radiation correction to temperature was applied following the method of Shaddix [19]. A simple energy balance on the thermocouple gives,

$$T_g = T_{tc} + \varepsilon_{tc} \sigma (T_{tc}^4 - T_w^4) \frac{d}{kNu} \quad (1)$$

Here, T_g , T_{tc} and T_w are, respectively, the true gas temperature at the point of measurement, the temperature measured by the thermocouple and the wall temperature. ε_{tc} , σ , k , d and Nu , respectively, are the emissivity of the thermocouple, the Stefan-Boltzmann constant, the thermal

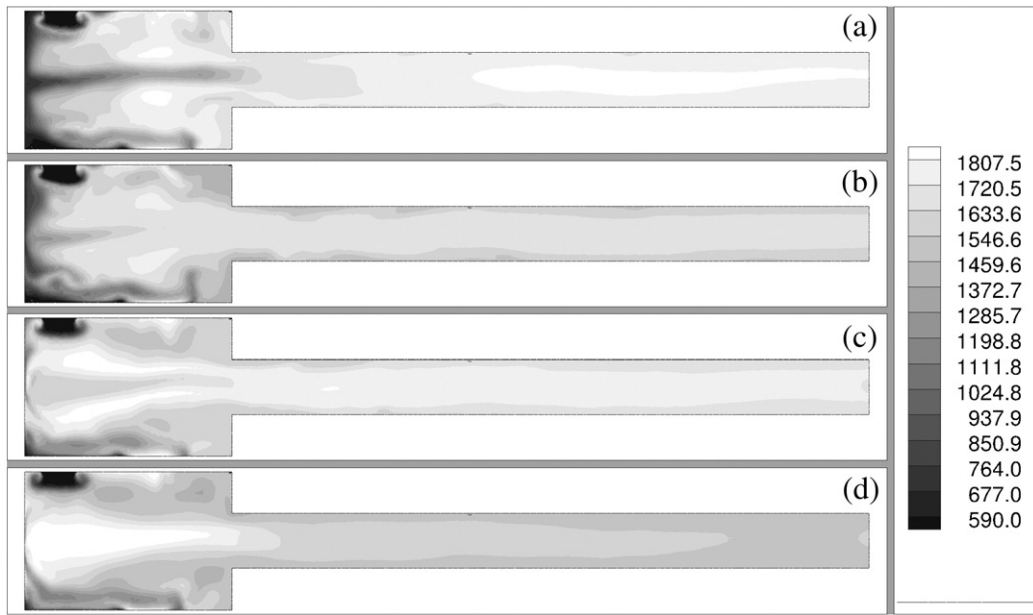


Fig. 14. Contours of temperature (°C) in the plane mid-x for λ values of (a) 1.06, (b) 1.21, (c) 1.31 and (d) 1.69.

conductivity of the gaseous medium, the thermocouple diameter and the Nusselt number based on d . For more details, the reader is referred to Shaddix [19]. The Nusselt number correlation [20] used herein is,

$$Nu = \left(0.24 + 0.56Re_d^{0.45}\right) \left(\frac{T_m}{T_\infty}\right)^{0.17} \quad (2)$$

The bulk gas velocity evaluated from the known fuel and air mass flow rates was used to calculate the Reynolds number Re_d based on d . The farfield temperature T_∞ was taken to be T_g , and the mean temperature T_m was taken to be the arithmetic mean of T_g and T_{tc} . Although one could come up with better choices for T_∞ and T_m , the dependence of T_m/T_∞ on Nu is rather weak, and the available information is put to use

in the best possible way in Eq. (2). Under the conditions of operation of the combustor, ε is approximated [21] to 0.8.

5.2.4. Comparison of CFD and experiment

Prior to comparing experimentally measured temperatures to CFD predictions, a temperature correction to account for radiative heat loss was applied, as discussed in 5.2.3. The measured wall temperature $T_w = 810$ °C was used in the calculation. The radiation-corrected temperatures are compared with corresponding CFD values in Fig. 16. The precision uncertainty in temperature quoted by the thermocouple manufacturer was 1% of the reading. The most significant source of error in temperature is expected to arise out of the uncertainty in the thermocouple emissivity. An assumed uncertainty of ± 0.2 is propagated to obtain the error estimates shown in Fig. 16.

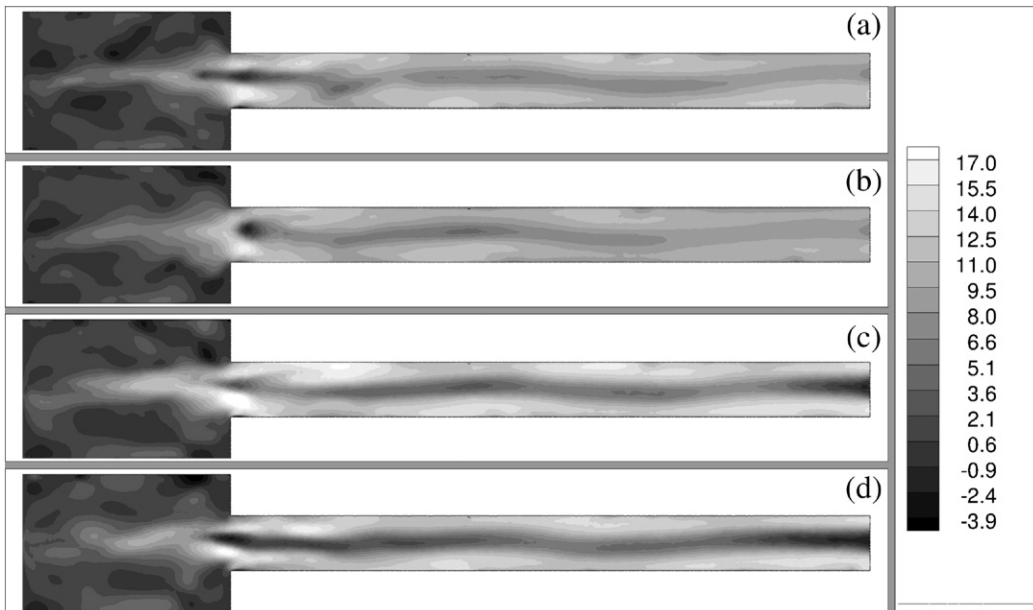


Fig. 15. Contours of axial-velocity (m/s) in the plane mid-x for λ values of (a) 1.06, (b) 1.21, (c) 1.31 and (d) 1.69.

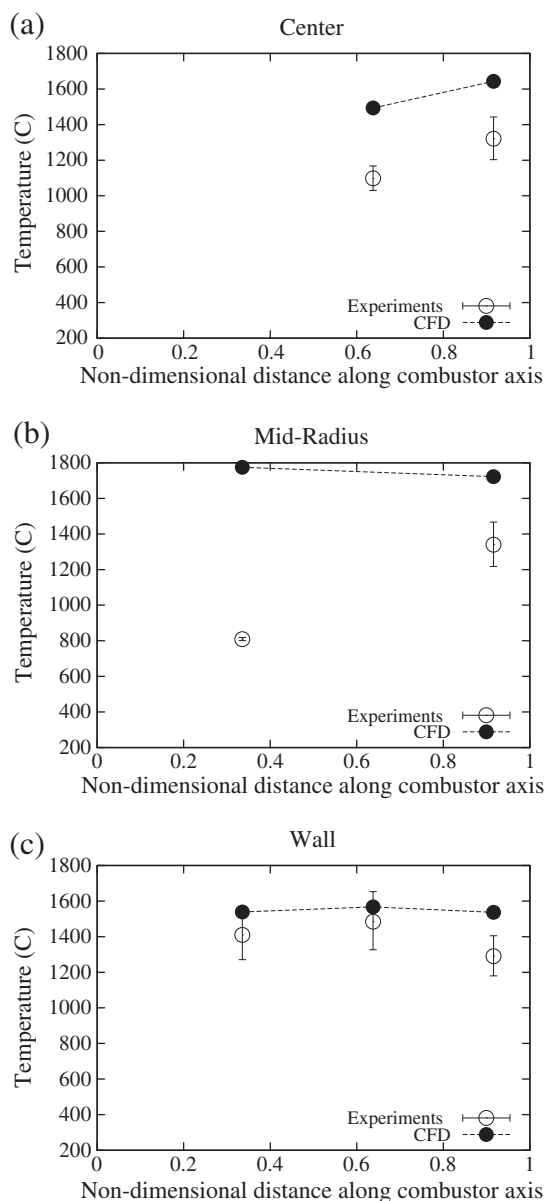


Fig. 16. Comparison of CFD with experiment for condition 1 in Table 3 ($\lambda = 1.21$). Shown here are temperatures at (a) the combustor axis, (b) 2/3 radius away from the wall and (c) 1/3 radius away from the wall.

Except for one outlier in Fig. 16(b), the experiments and simulations follow each other fairly closely. It is noteworthy that the agreement between CFD and experimental data is less along the combustor centreline, and improves as one moves closer to the wall. The highest turbulent activity being expected along the combustor axis, it is plausible that the heart of the reaction zone vigorously oscillated about the points of measurement, causing a low average temperature to be recorded. Closer to the wall, however, a relatively more steady flame rendered the measured temperatures more accurate.

The use of thick thermocouple stems necessitated by the harsh environment within the combustor posed a challenge in recording accurate temperatures. CFD, on the other hand, was expected to give good predictions of the thermal field. Owing to the low gas speeds in the combustor, as seen in Fig. 15, the chemical reactions in the flame were not expected to be rate-limited, and the equilibrium chemistry approximation discussed in Section 4.3 was deemed adequate for the purpose of obtaining the temperature field.

6. Conclusion

The mixing and burning characteristics of a cyclone combustor in development have been studied both experimentally and numerically. The gases entering through the fuel and air streams were seen to mix well before exiting the combustor. Based on the mixing characteristics in CFD, the Reynolds Stress Model for turbulence was deemed to be suitable in this application. CFD predictions of temperature were consistently higher than measured values in the mixing studies; this is primarily a consequence of the higher heat capacity of the combustion products of the gas engine in comparison to that of air.

The combustion studies indicated a stable flame resulting in complete combustion of particulate and tar laden producer gas. Particulate emissions measurements in the flue gas show that the combustion process adequately complies with applicable emissions norms. The agreement between experimentally measured temperatures and those predicted by CFD is good near the combustor walls, and deteriorates as one approaches the combustor axis. Higher gas speeds and turbulence activity along the axis could account for this behaviour.

Acknowledgement

The entire work reported in this paper was carried out at Siemens Renewable Energy Innovation Centre, Bangalore and authors are thankful to Siemens Corporate Research and Technology Management for supporting this work.

The authors also wish to acknowledge the contributions of Arun Girimaji, Vinayak Kulkarni and Raghu in conducting the experimental trials, and also those of former employees Aaditya Uppal, Chandrasekhar Joshi and Vijay Kumar Pantangi towards this project.

References

- [1] H.S. Mukunda, *Understanding clean energy and fuels from biomass*, Wiley India Pvt. Ltd., 2011.
- [2] A.G. Bhawe, D.K. Vyas, J.B. Patel, A wet packed bed scrubber-based producer gas cooling-cleaning system, *Renewable Energy* 38 (2008) 1716–1720.
- [3] K. Kandassamy, E. Natarajan, S. Renganarayanan, Producer gas cleaning techniques, *Proceedings of the International Conference on Fluidized Bed Combustion*, 2003.
- [4] S. Dasappa, P.J. Paul, H.S. Mukunda, N.K.S. Rajan, G. Sridhar, H.V. Sridhar, Biomass gasification technology – a route to meet energy needs, *Current Science* 87 (7) (2004) 908–916.
- [5] H.J. Sheen, W.J. Chen, S.Y. Jeng, T.L. Huang, Correlation of swirl number for a radial-type swirl generator, *Experimental Thermal and Fluid Science* 12 (1996) 444–451.
- [6] S. Turns, *An introduction to combustion: concepts and applications*, McGraw-Hill, 2011.
- [7] A. Parihar, C. Joshi, G. Sridhar, The performance of cyclones in producer gas cleaning: Experimental and modeling studies, *Proceedings of the Institution of Mechanical Engineers Part A: Journal of Power and Energy* 226 (6) (2012) 1–18.
- [8] M.J.A.D.B. Kittelson, W.F. Watts, Review of diesel particulate matter sampling methods, Final report, USEPA, 1998.
- [9] S.B. Pope, *Turbulence*, Cambridge University Press, 2000.
- [10] T. Poinso, D. Veynante, *Theoretical and Numerical Combustion*, Edwards, 2005.
- [11] T.L. Bergman, A.S. Lavine, F.P. Incropera, D.P. Dewitt, *Fundamentals of heat and mass transfer*, John Wiley & Sons, New York, 2007.
- [12] S.W. Churchill, H.H.S. Chu, Correlating equations for laminar and turbulent free convection from a vertical plate, *International Journal of Heat and Mass Transfer* 18 (11) (1975) 1323–1329.
- [13] S.W. Churchill, H.H.S. Chu, Correlating equations for laminar and turbulent free convection from a horizontal cylinder, *International Journal of Heat and Mass Transfer* 18 (9) (1975) 1049–1053.
- [14] H. Shalaby, K. Pachler, K. Wozniak, G. Wozniak, Comparative study of the continuous phase flow in a cyclone separator using different turbulence models, *International Journal for Numerical Methods in Fluids* 48 (2005) 1175–1197.
- [15] F. Kaya, I. Karagoz, Performance analysis of numerical schemes in highly swirling turbulent flows in cyclones, *Current Science* 94 (10) (2008) 1273–1287.
- [16] B. Wang, D.L. Xu, G.X. Xiao, K.W. Chu, A.B. Yu, Numerical study of gas-solid flow in a cyclone separator, *Third International Conference on CFD in the Minerals and Process Industries*, Melbourne, Australia, 2003.
- [17] S.M. Fraser, A.M. Abdel-Razek, M.Z. Abdullah, Computational and experimental investigations in a cyclone dust separator, *Proceedings of the Institution of Mechanical Engineers Part E: Journal of Process Mechanical Engineering* 211 (1997) 247–257.
- [18] B. Sengupta, *Environmental standards for ambient air, automobiles, fuels, industries and noise*, Pollution control law series PCLS/4/2000-2001, Central Pollution Control Board, New Delhi, India, 2000.

- [19] C.R. Shaddix, Correcting thermocouple measurements for radiation loss: A critical review, Proceedings of the 33rd National Heat Transfer Conference, Albuquerque, New Mexico, 1999.
- [20] D.C. Collis, M.J. Williams, Two-dimensional convection from heated wires at low reynolds numbers, *Journal of Fluid Mechanics* 6 (1959) 357–384.
- [21] G.A. Greene, C.C. Finfrock, T. Irvine, Total hemispherical emissivity of oxidized inconel 718 in the temperature range 300 – 1000°C, *Experimental Thermal and Fluid Science* 22 (2000) 145–153.